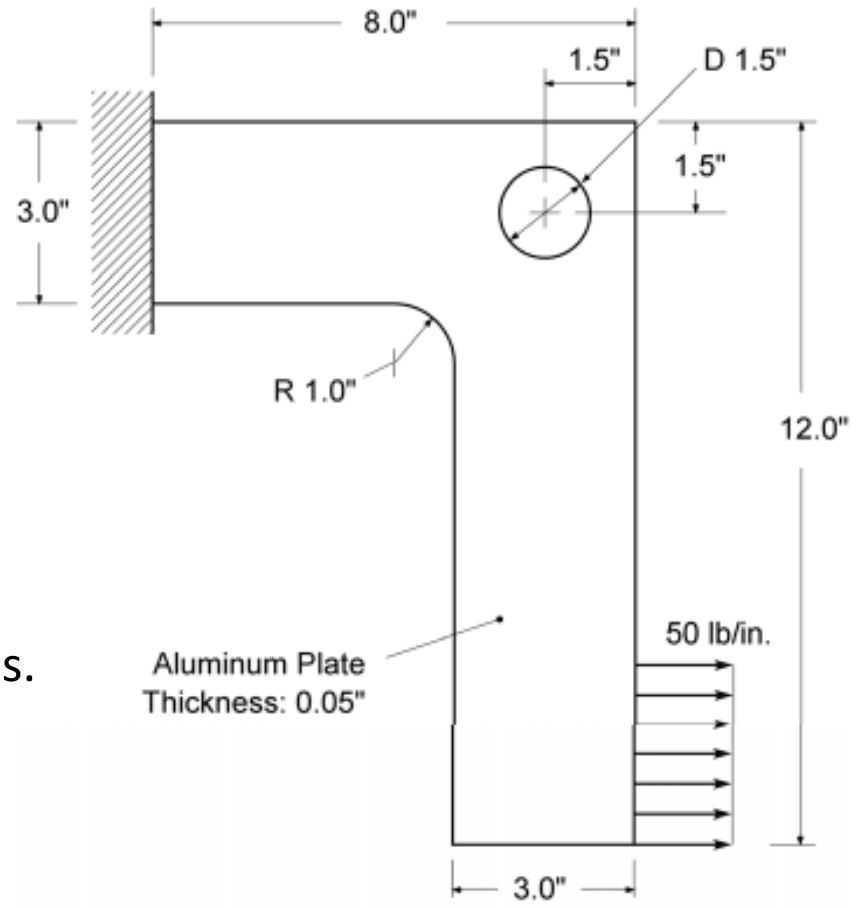
ABAQUS/CAE Tutorial: Analysis of an Aluminum Bracket

By Hyonny Kim

Last updated July 2010



In this tutorial, you'll learn how to:

1. Sketch 2D geometry & define part.
2. Define material properties.
3. Apply loads and boundary conditions.
4. Mesh.
5. Run analysis.
6. View results.

Helpful Tips Before Getting Started

HELP

Online help manuals: *abaqus_aae doc & - there is a "book" for CAE: "ABAQUS/CAE User's Manual". Context* sensitive help is also available within CAE.

CAE creates the .inp file which you can edit and run by the command line, or you can submit jobs from within CAE. Other files are .cae (CAE model file), .odb, .dat, .log, .msg, and .sta. The .dat is the text output file that will contain results. The .odb file is the binary output file that will be read during post-processing to view graphical results. The .log file keeps a text record of all processes and is useful for checking the status of the analysis. The .msg lists the progress of the analysis, as well as provides some messages about why an analysis might have crashed (this information is often within the .dat file as well). The .sta file is a summary of the information contained in the .msg file, and is useful for monitoring the status of long-running jobs during their computation.

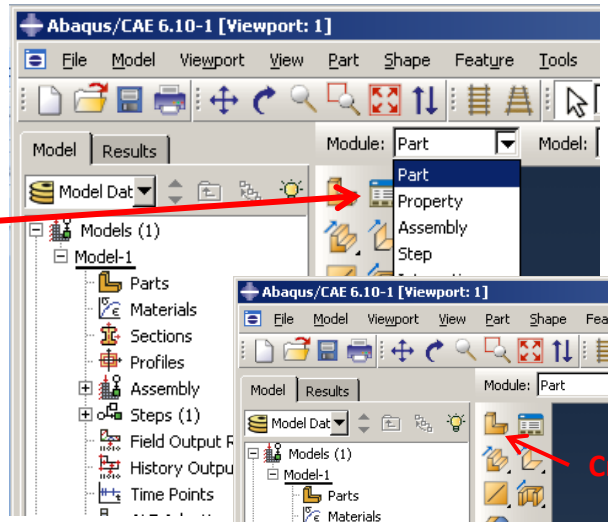
MOUSE

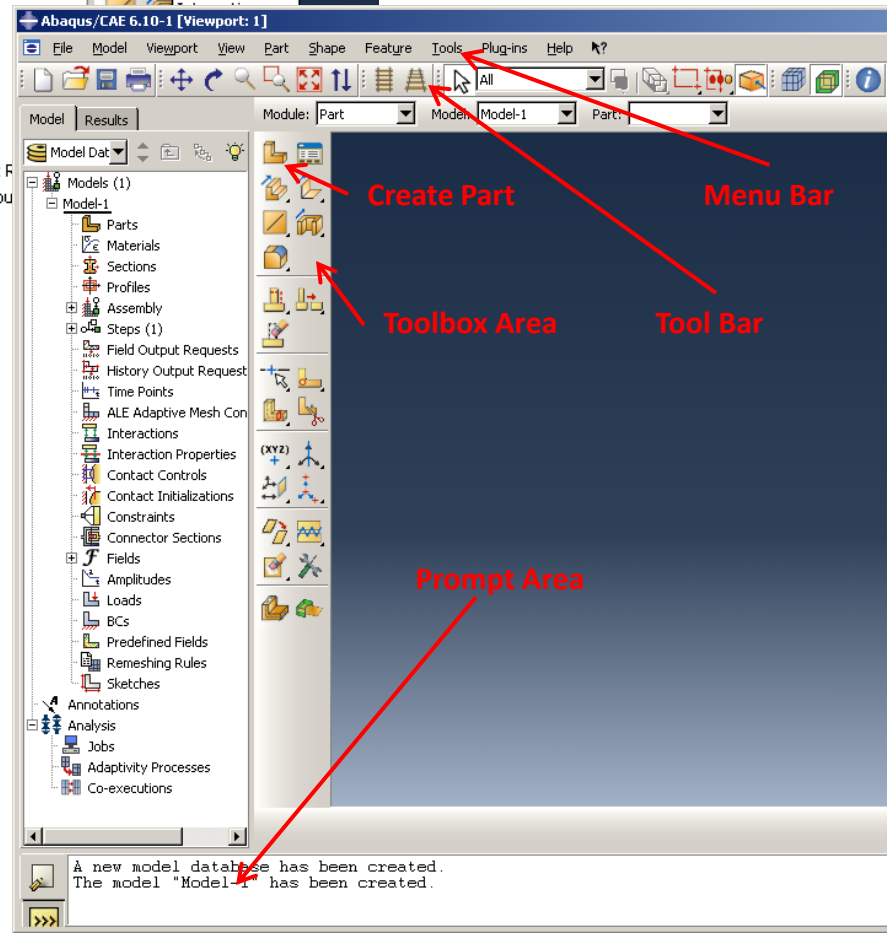
Use of the Mouse:

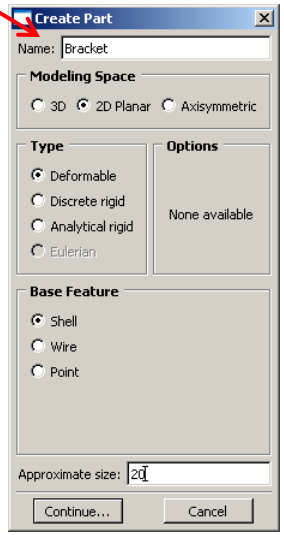
- button 1 (left) is select, button 2 (right) gives menu, button 3 (middle, if available is "enter" or "done")
- multiple items can be selected by: "dragging" a window or holding the SHIFT key while picking
- items can be de-selected by holding the CTRL key.

ABAQUS/CAE: Getting Started, Create Part

- Open Abaqus/CAE
- Click **Create Model Database With Standard/Explicit Model**
- In the **Module dropdown box,** select **Part**
- Note the locations of: **Tool Bar, Menu Bar, Toolbox Area, Prompt Area.** These will be referred to repeatedly in the future.
- In the *Toolbox Area,* click **Create Part** button. The *Create Part* window will pop up.
- Enter in name, e.g., *bracket*
- Under *Modeling Space,* choose **2D Planar**
- *Base Feature*, **Shell**
- *Approximate size:* **20**
- Click **Continue…**







Sketch Part

- The window will change to that shown at right.

  Note the buttons pointed out.

  – Buttons with a dark triangle will provide more button choices when clicked and held. Try it.

  – Float your mouse pointer over buttons, it will give a pop-up description.

  – Context-Sensitive Help is available. Click the help button, then the item you want more info on.

1. Click **Create Lines** button. Note it is highlighted when active. In prompt area, enter in the coordinates:

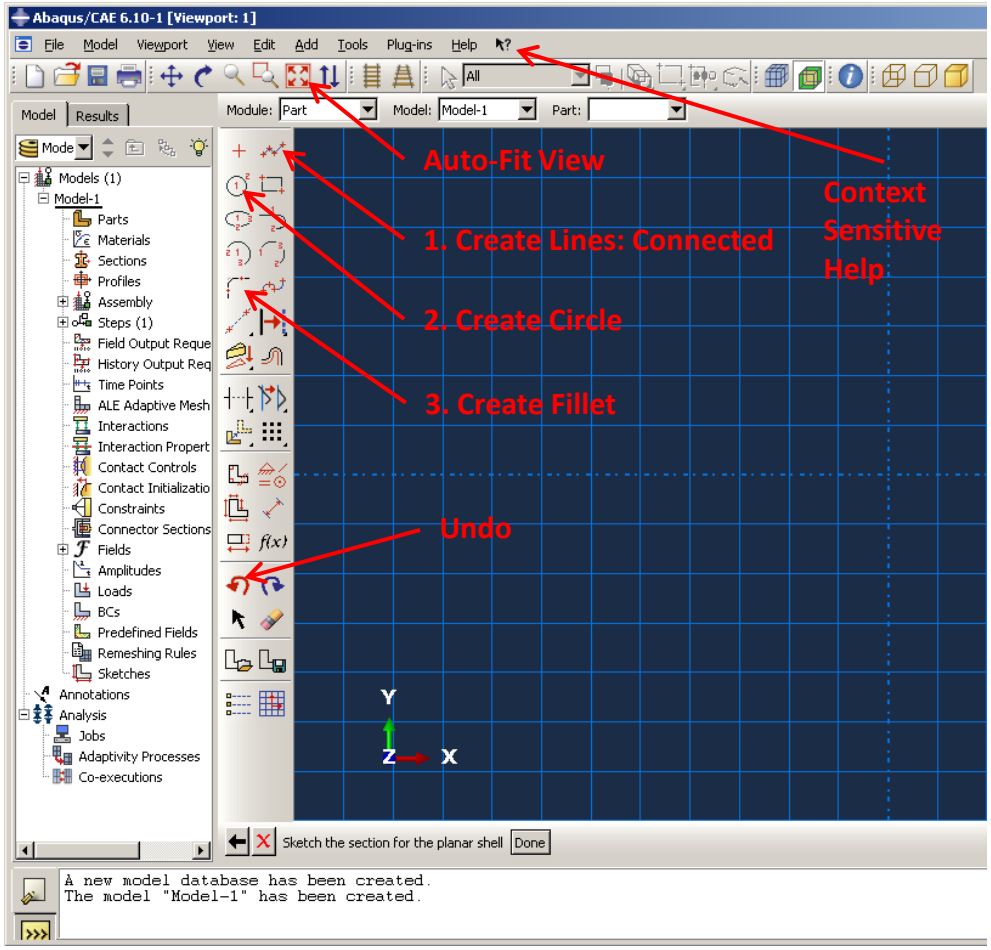
   (0, 0), (8,0), (8,-12), (5,-12), (5,-3), (0,-3)

   Click on point 1 (box will appear on it) Finished product will be yellow outline of the bracket.

   Click **Auto-Fit** View button to re-scale image. The other buttons adjacent to this one will adjust panning, rotation, and zoom. Try them out. Dynamic viewing with mouse buttons by holding CTRL + ALT on right side of keyboard. Try it.

2. Click **Create Circle** button. Enter (6.5, -1.5) for *center,* and *(7.25, -1.5)* for perimeter point.

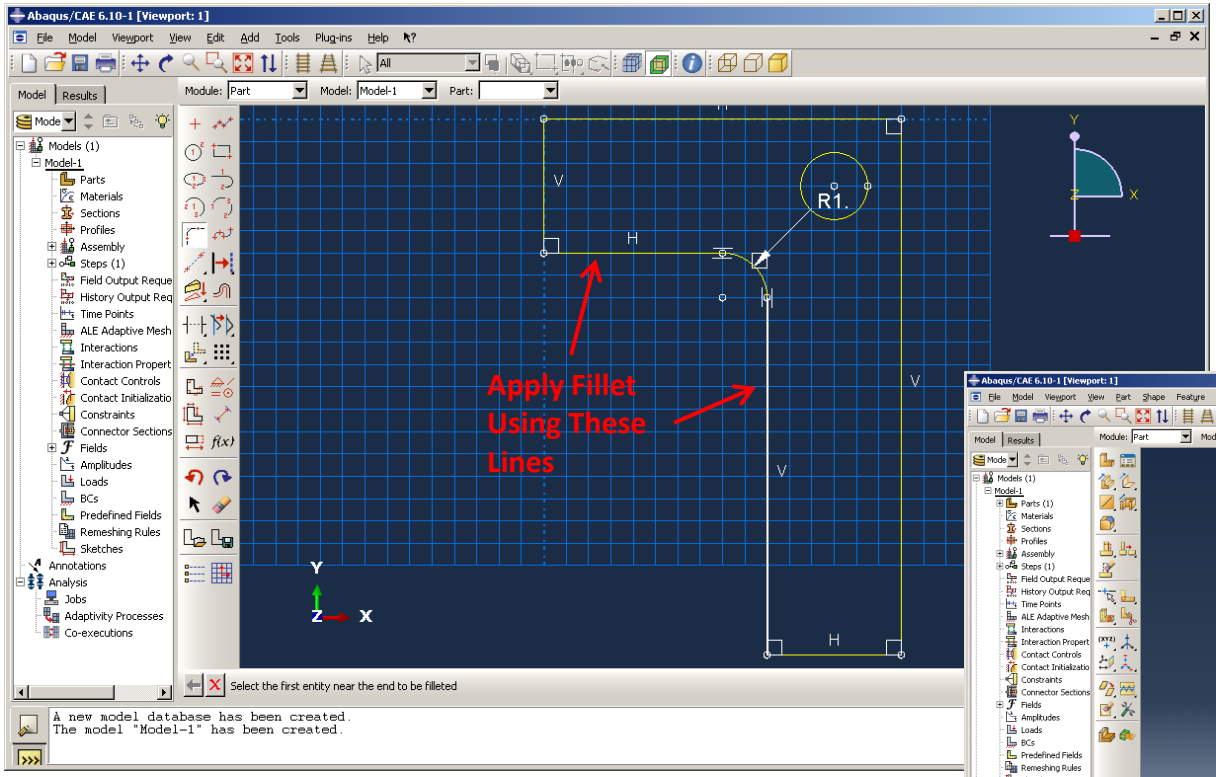
3. Click **Create Fillet button**… (go to next page)

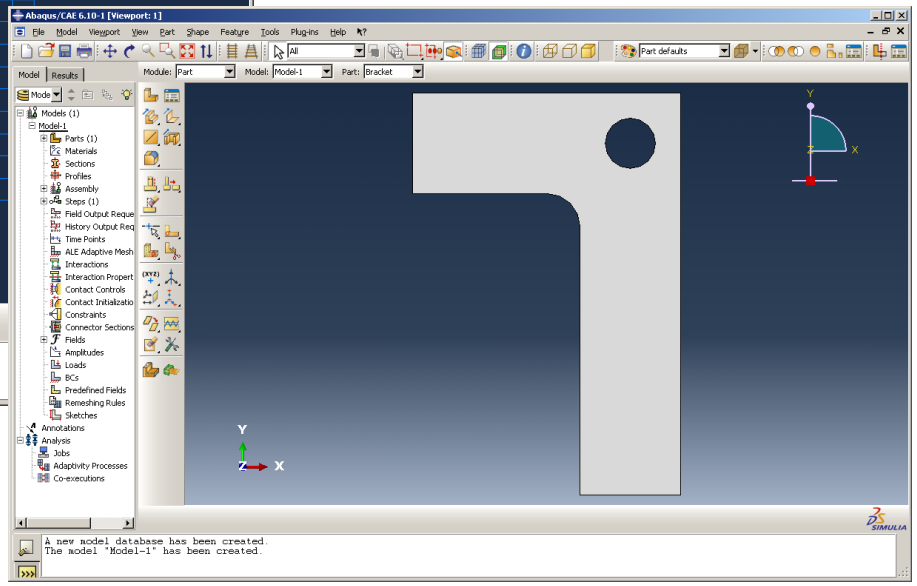


Sketch Part Continued

3. continued… enter **1.0** for *fillet radius* in the *Prompt Area,* hit enter, then Mouse click on inner two lines when prompted.

- The **Create Fillet** button should still be highlighted. Click this again to get the screen shown below left.
- Click the **Done** button in the **Prompt Area** at the bottom of the screen.
- You now are returned to the **Part Module** screen. This should look like the screen below right. Note different tool buttons shown in the *Toolbox Area.*

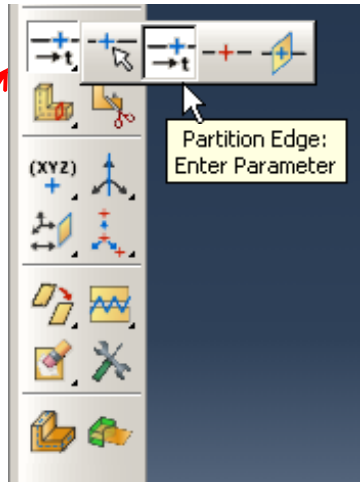

Abaqus/CAE 6.10-1 [Viewport: 1]
R1.
Apply Fillet Using These Lines
Select the first entity near the end to be filleted
A new model database has been created.
The model "Model-1" has been created.

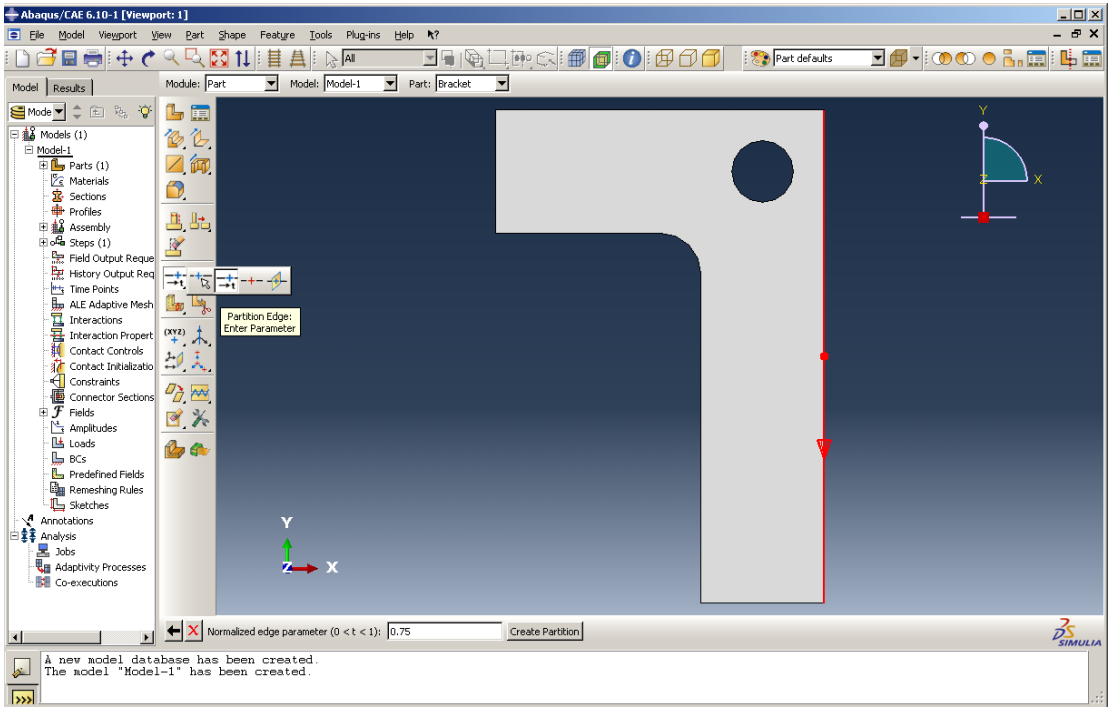


Partition Edge

- Click the "**Partition Edge: Enter Parameter"** button. In order to get this button, you need to click and hold over the line partitioning tools button – note the small dark triangle in the lower right corner indicates that this is an expanding button.
- You will be prompted to select an edge, select the far right-hand edge of the bracket.
- Click **Done.**
- In the Prompt Area, enter in value of **0.75 for the** *Normalized edge parameter.*
- Click the **Create Partition** button to finish.
- You will see a large dot one fourth of the way up the edge of the bracket. This partitioned edge will be used later for applying a uniform load.

Line Partitioning Tools-Click and Hold



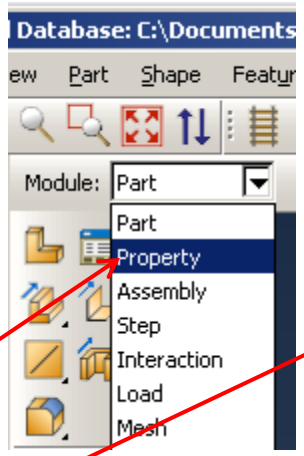

Abaqus/CAE 6.10-1 [Viewport: 1]
Partition Edge:
Enter Parameter
Normalized edge parameter (0 < t < 1):
0.75
Create Partition
A new model database has been created.
The model "Model-1" has been created.

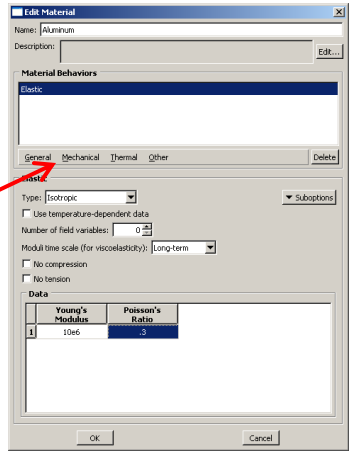
Saving and Defining Material Properties

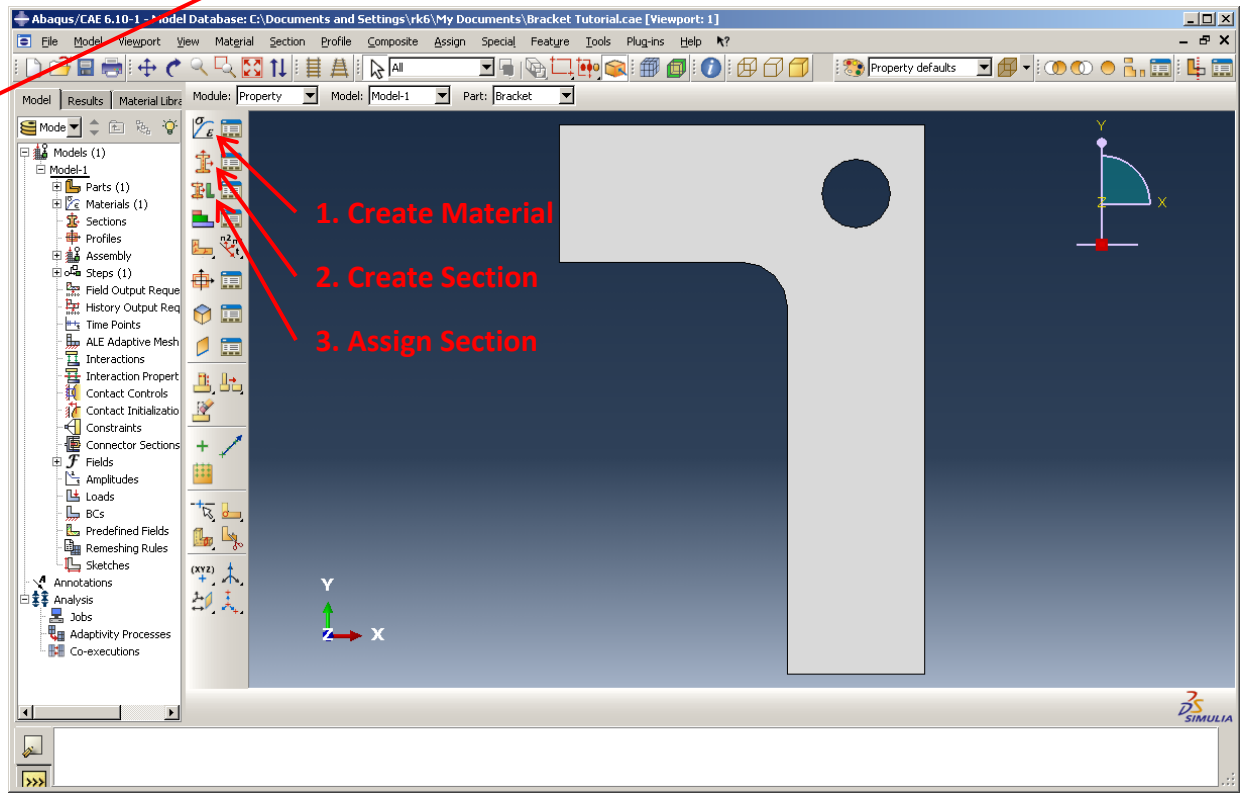
- Save your work: in the **Menu Bar**, click **File, Save As.**
  - – Under *Selection,* enter a name at the end of the path, e.g. *bracket.* Click ***OK.*** From now on, you can just click the **blue floppy disk** icon in the **Tool Bar.** ***Save often!!!***
- Change Modules. In the **Module** drop-down box beneath the **Tool Bar**, select **Property.**

1. Click **Create Material Button**
   - – enter a name, e.g. *Aluminum,* select **Mechanical --> Elasticity --> Elastic**
   - – enter **10e6** for *Young's Modulus,* **0.3** for *Poisson's Ratio.*
   - – click **OK**

- If you want to modify the material, click the **Material Manager** button to the right of **Create Material,** select the material by name and click **Edit**, or click **Dismiss** to leave without making any changes

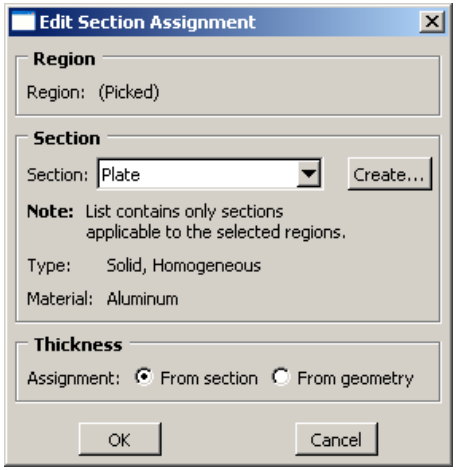


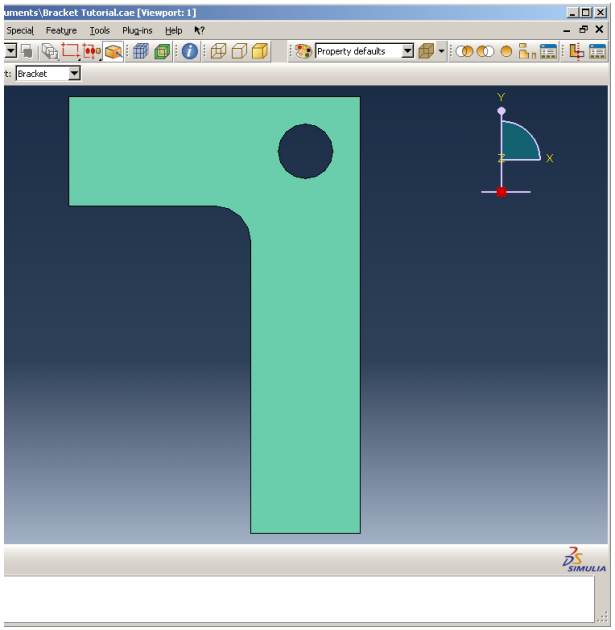




Assign Properties to Regions of Model

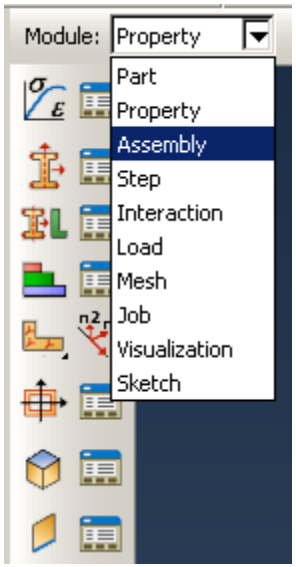
2. Click **Create Section Button**
   - enter a name, e.g., *plate*
   - choose **Solid and Homogeneous**
   - click **Continue**
   - select the material *Aluminum* (or what ever you named it, there should be only one to choose from) in the drop down box
   - enter *Plane stress/strain thickness:* ***0.05.*** Click ***OK.***
3. Click **Assign Section** Button
   - you will be prompted to select a region. **Click on the part.**
   - Click the **Done** button in the **Prompt Area** at the bottom of the screen.
   - The *Section Assignment* window will pop up. Select the *Section Name* you wish to assign to this region (there should be only one which you've previously named, e.g., *plate)*
   - click **OK.**

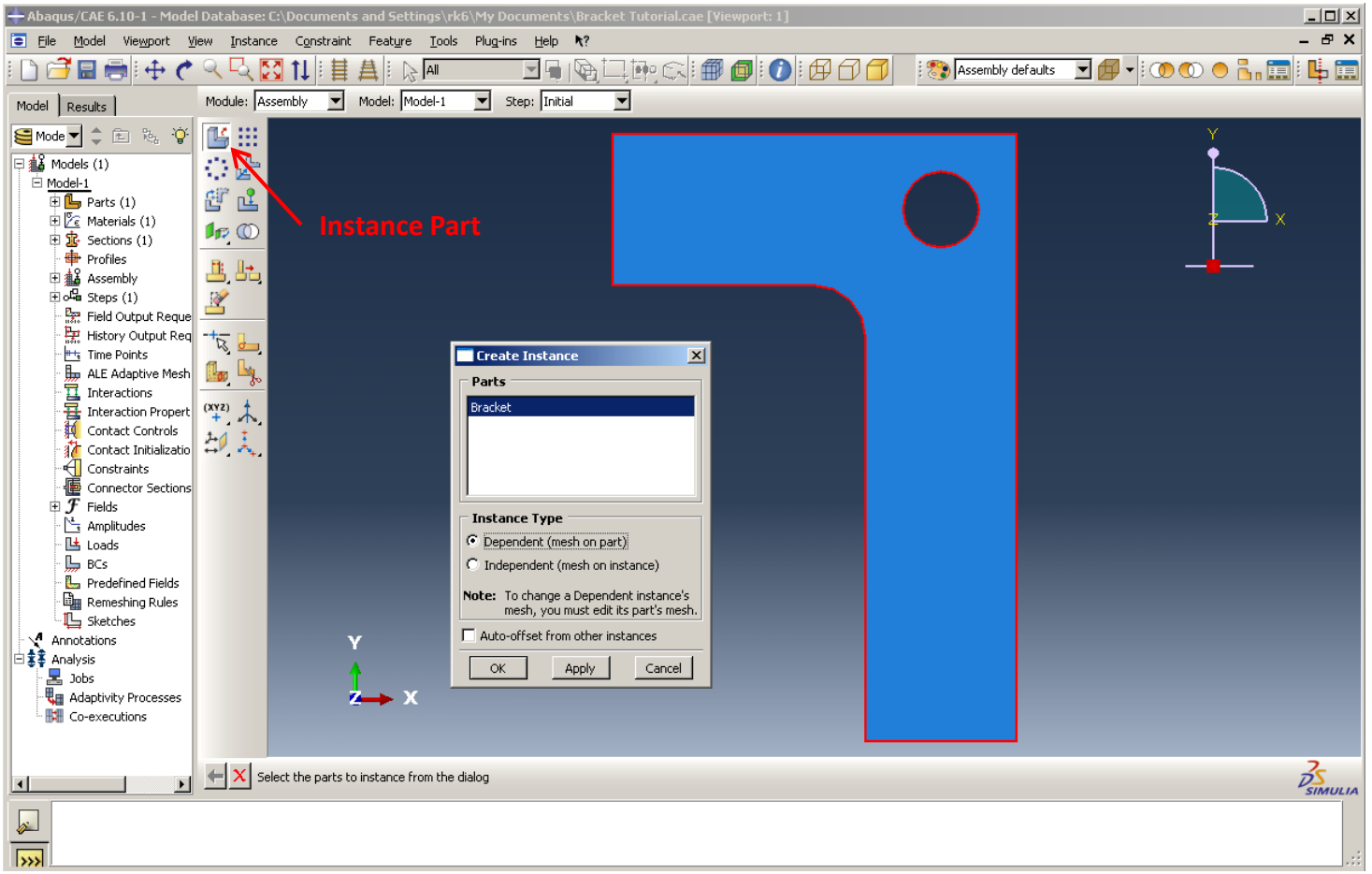




Instance Part

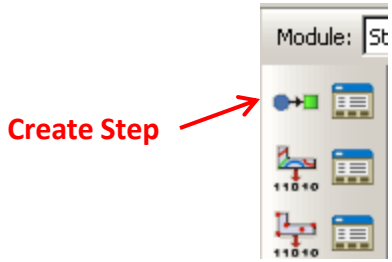
- Change Modules. In the **Module drop-down box** beneath the **Tool Bar,** select the **Assembly.**
  - note, the *Canvas* (main working graphical window) will be blank.
- Click the **Instance Part** button. The *Create Instance* window will pop up.
- Select the part you wish to instance, e.g., the part we named *bracket* previously. A red outline of the bracket will show. Choose *Independent* as the Instance Type
- Click **OK.**

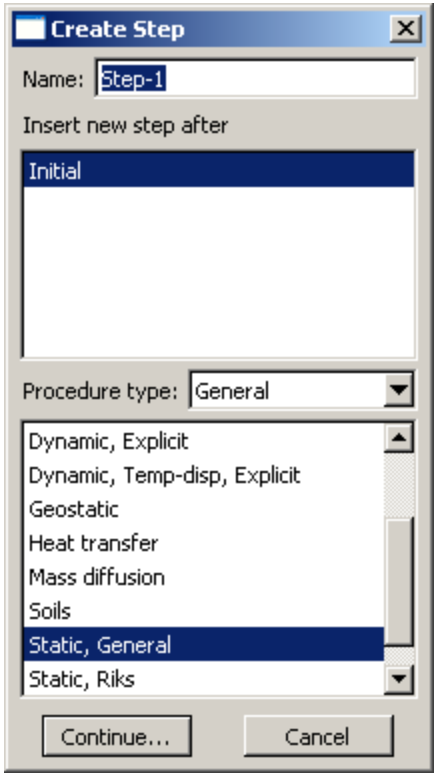


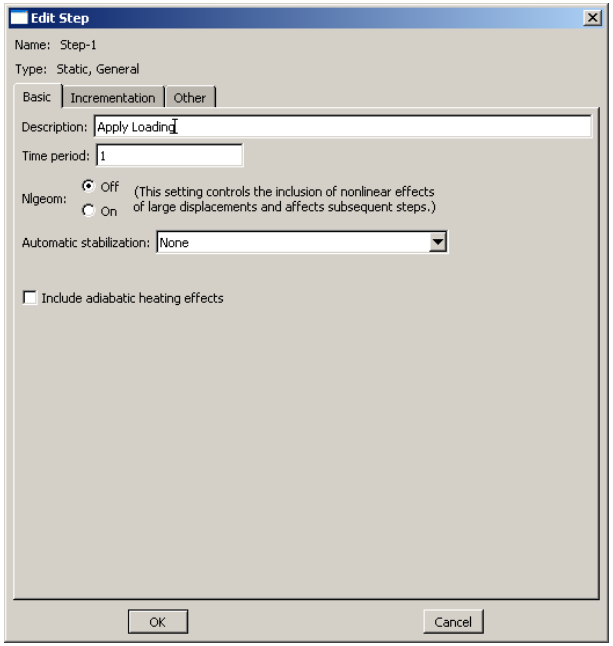


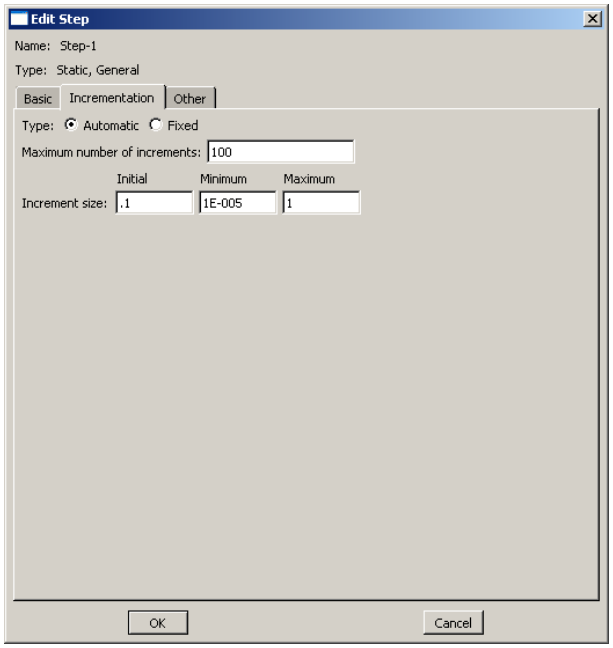
Step

- Change Modules. Select the **Step module.**
- Click the **Create Step** button.
  - *Create Step* window pops up
  - enter a name – use the default name *Step-1.*
  - be sure *Procedure type is set* to **General**, and **Static, General** is highlighted in the list. Click **Continue.**
- *Edit* Step window pops up, with the *Basic* tab active.
  - enter in a *Description,* e.g., *apply loading*
- Click the *Incrementation tab.*
  - under *Increment Size, enter* value of 0.1 for *Initial. Leave* the rest the same. Full load corresponds to an Increment value of 1 (when *Time Period* is set to 1 under the *Basic tab).* Setting *Initial to* 0.1 forces Abaqus to start the analysis by applying 1/10 of the full load. This can also be left to default 1 value and the software will auto-select.
  - click **OK.**



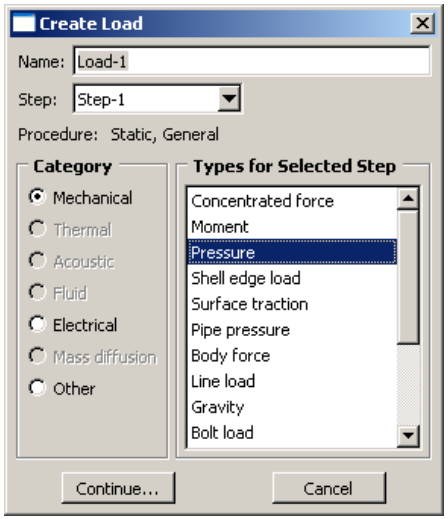


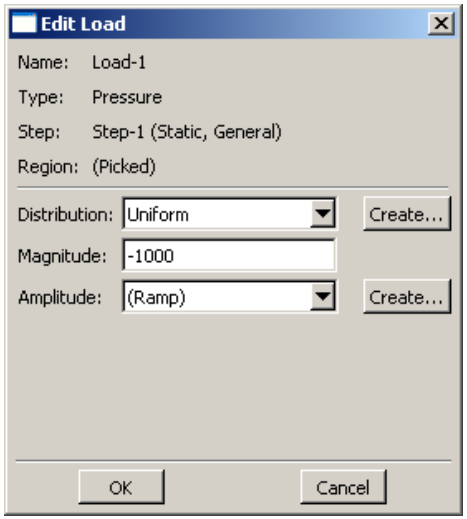


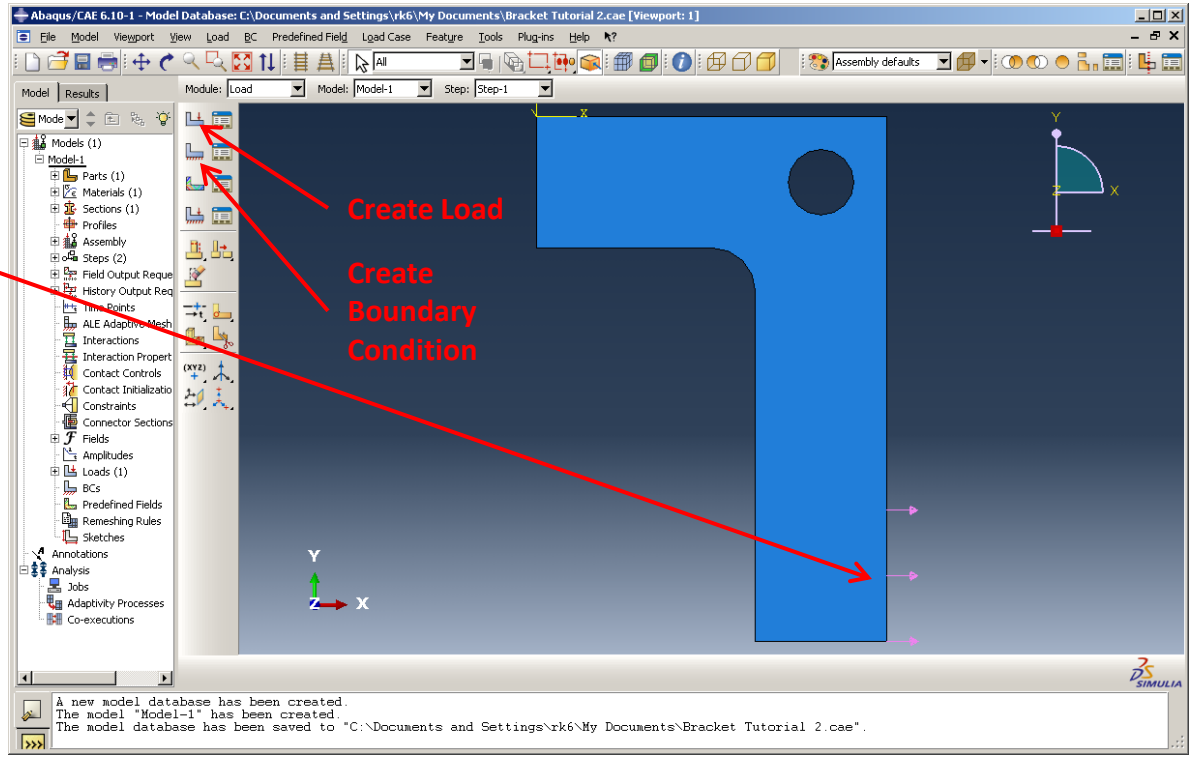


Load

- Change Modules. Select the **Load** module.
- Click the **Create Load** button.
  - – the *Create Load* window pops up.
  - – enter a *Name*, e.g., *Load-1* is the default name
  - – be sure **Step-1** (or what name you have given it) is selected in the *Step* drop down box.
  - – be sure **Mechanical** is selected in *Category*
  - – under *Type for selected step,* choose **Pressure**
  - – click **Continue**
  - – Upon prompting to *Select surfaces,* mouse pointer click on the lower portion of the right edge of the bracket, the region we partitioned previously. It will highlight red.
  - – click **Done.**
  - – in the *Edit Load* window that pops up, be sure to have *Distribution* set to **Uniform**, enter value of **–1000** in *Magnitude*, and be sure that **(Ramp)** is selected under *Amplitude.* This is a 1000 psi traction.
  - – click **OK.**
  - – You should get the image shown to the right. If your arrows are in the wrong direction, you need to go back and be sure to specify a *negative* pressure.

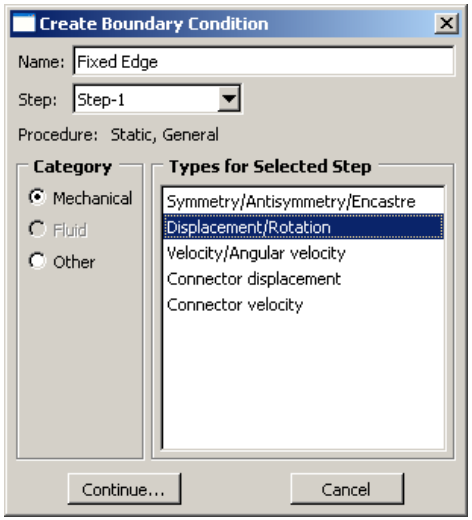


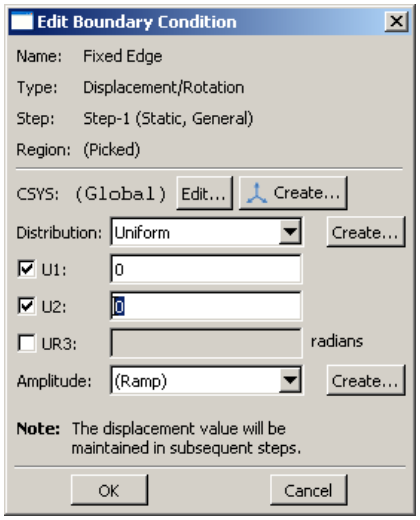


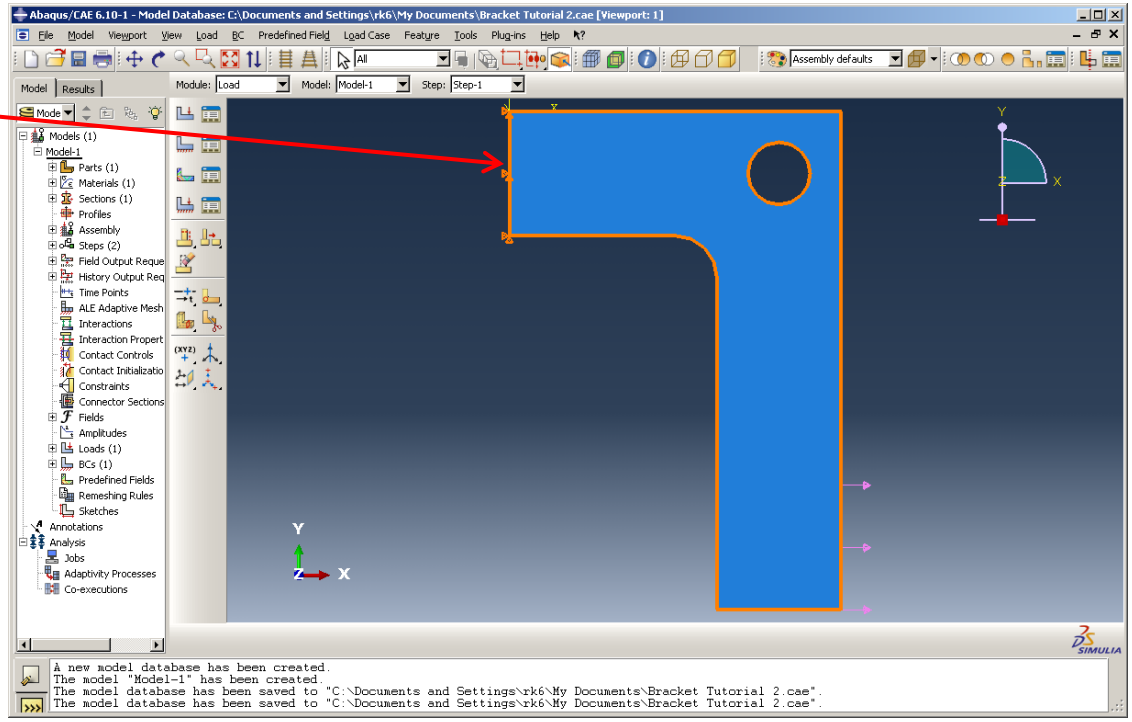

Abaqus/CAE 6.10-1 - Model Database: C:\Documents and Settings\rk6\My Documents\Bracket Tutorial 2.cae [Viewport: 1]
Create Load
Create Boundary Condition
A new model database has been created.
The model "Model-1" has been created.
The model database has been saved to "C:\Documents and Settings\rk6\My Documents\Bracket Tutorial 2.cae".

Boundary Conditions

- Click the **Create Boundary Condition** button.
  - the *Create Boundary Condition* window pops up.
  - enter a name, e.g., *fixed edge*
  - under *Category*, be sure that **Mechanical** is selected.
  - under *Type for selected step*, choose **Displacement/Rotation**
  - click **Continue.**
  - upon prompt to select regions, mouse pointer click on the upper left vertical edge of the bracket. It will highlight red.
  - click **Done.**
  - *Edit Boundary Condition* window pops up.
  - be sure **Uniform** is selected in *Distribution.*
  - check-mark (click) boxes for **u1** and **u2,** and leave the default values of 0.
  - click **OK.**
  - You should get the image shown to the right

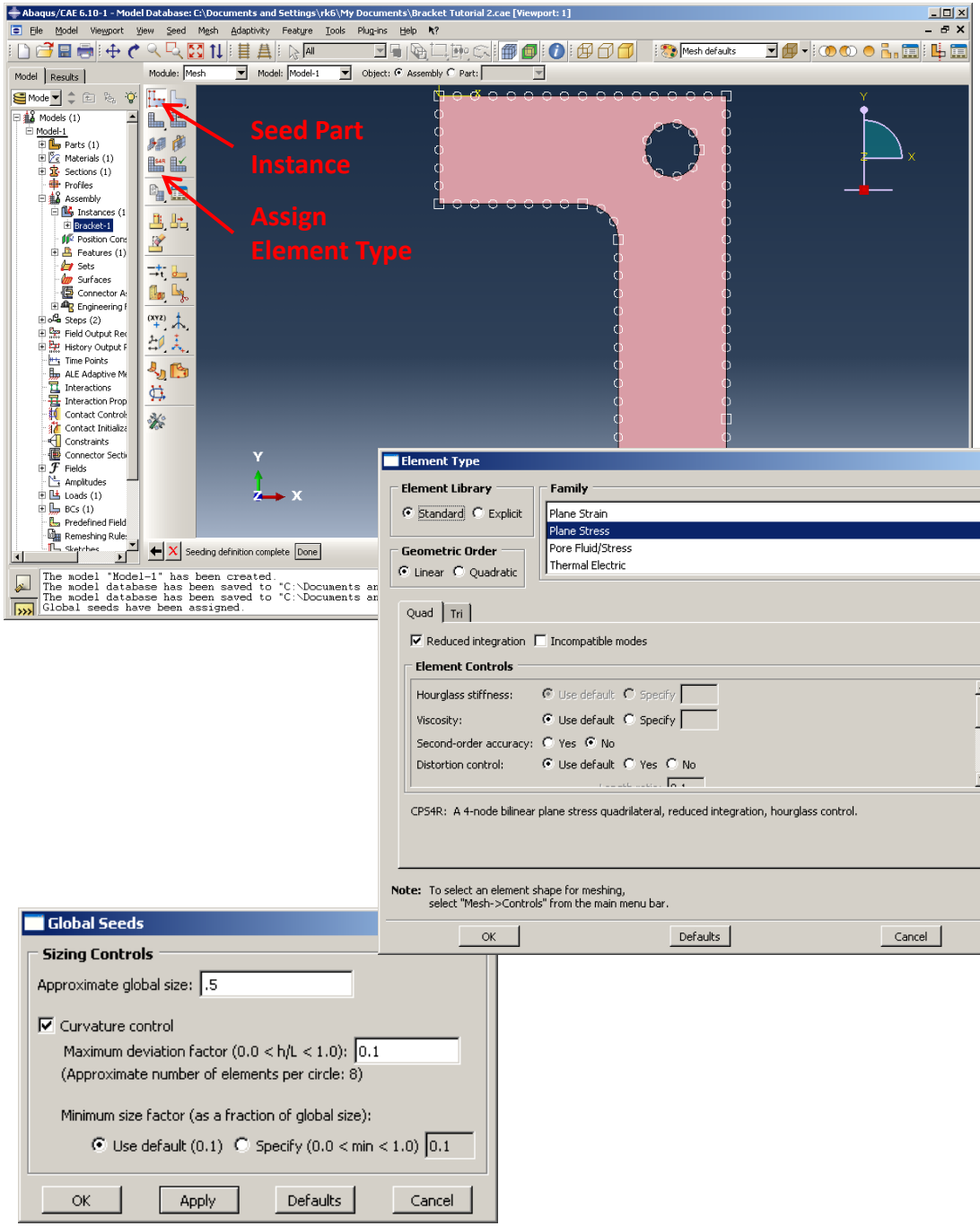






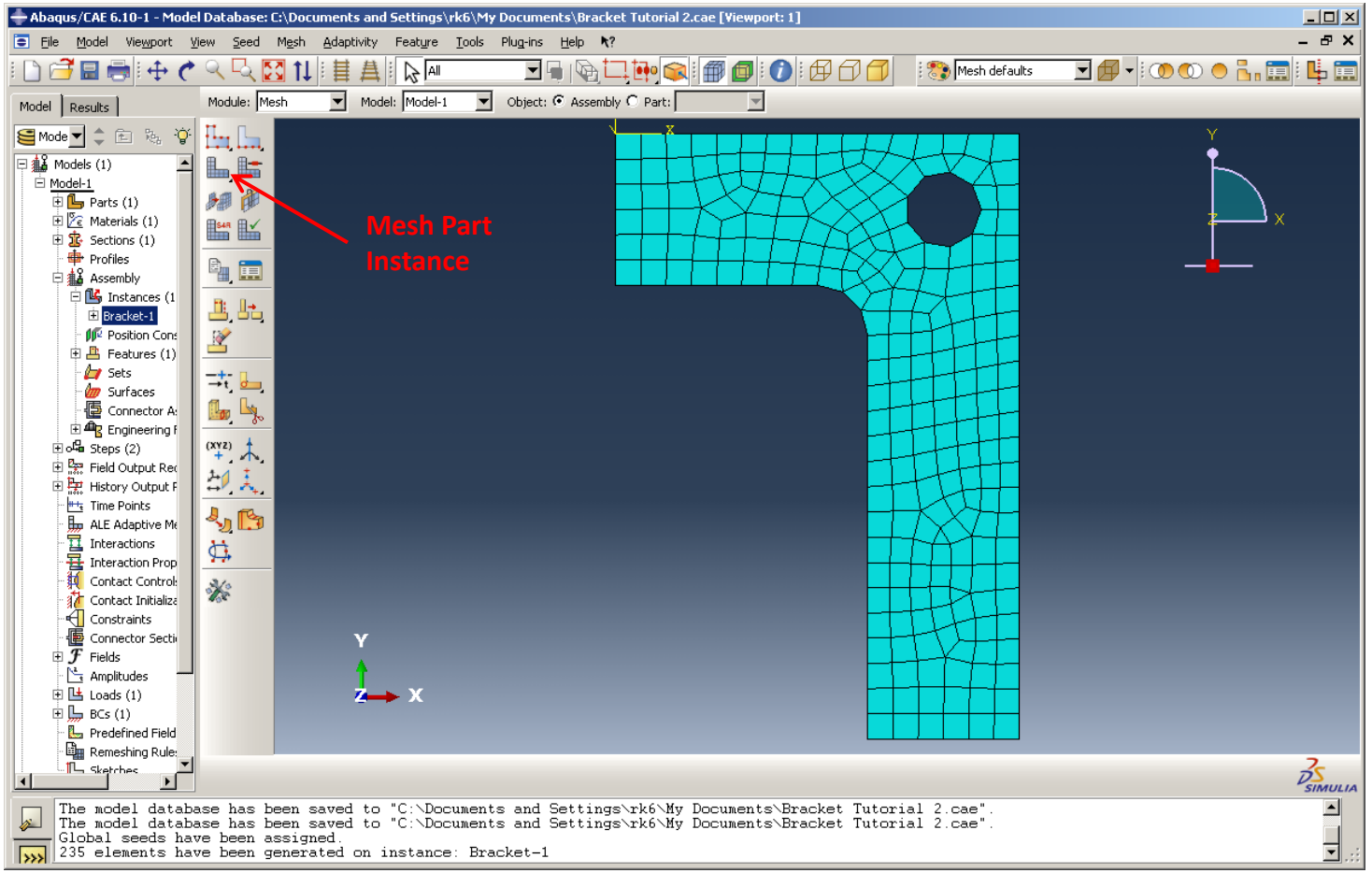
Seed Mesh

- Change Modules. Select the **Mesh** module.
- Click the **Seed Part Instance** button. This is an expandable button. There are many other functions within this button that are useful for controlling mesh size.
  - In the *Prompt Area,* enter a *Global element size value of **0.5.***
  - Hit enter and you will see circular symbols indicating nodal locations along the part edges.
- Click the **Assign Element Type** button.
  - the *Element Type* window pops up.
  - choose **Standard** in *Element Library*
  - **Plane Stress** in *Family*
  - **Linear** in *Geometric Order*
  - within the *Quad tab,* choose **Reduced Integration**. Leave everything else at default values.
  - the text in the lower box should indicate a *CPS4R element* identification. This is a 4-node reduced integration quadrilateral element.
  - click **OK.**



Mesh

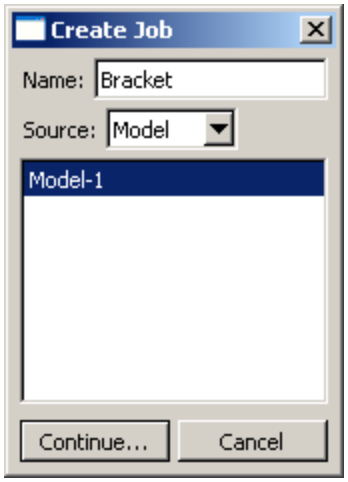
- Click **Mesh Part Instance** button. Note this button has many other functions within it (click-hold mouse button down on this button) such as deleting mesh and meshing regions of a part.
- Click **Yes** in the *Prompt Area.*
- Your mesh should look like the image shown to the right.

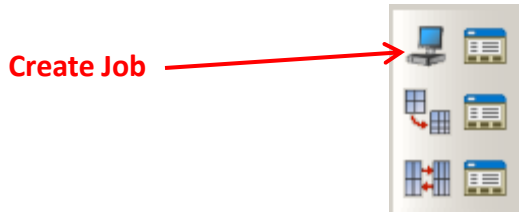


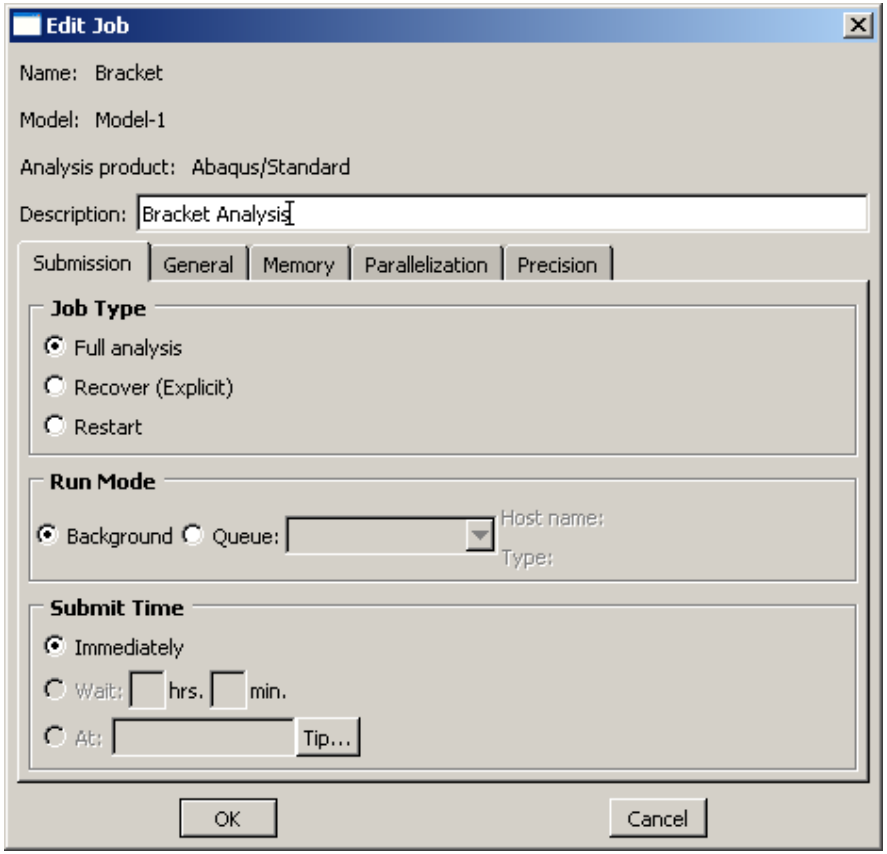
Create Job

SAVE YOUR WORK!!!

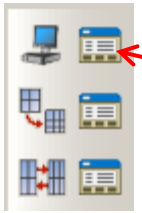
- Change Modules. Select the **Job** module.
- Click **Create Job** button.
- Enter a name, e.g., *bracket*
- Click **Continue.**
- In the *Edit Job* window that pops up, enter a *Description*, e.g., *bracket analysis*
- Check that **Full analysis**, **Background**, and **Immediately** are selected.
- Click **OK.**





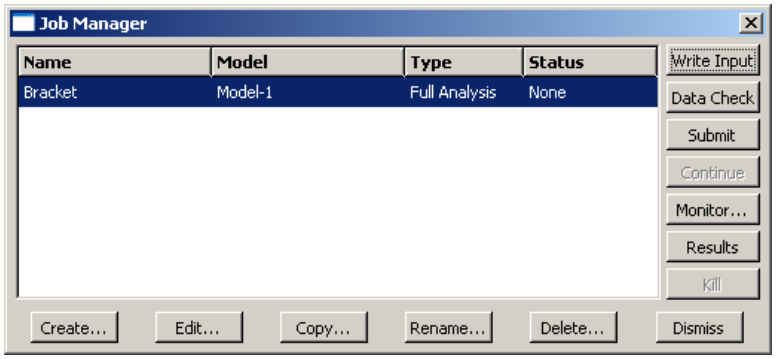


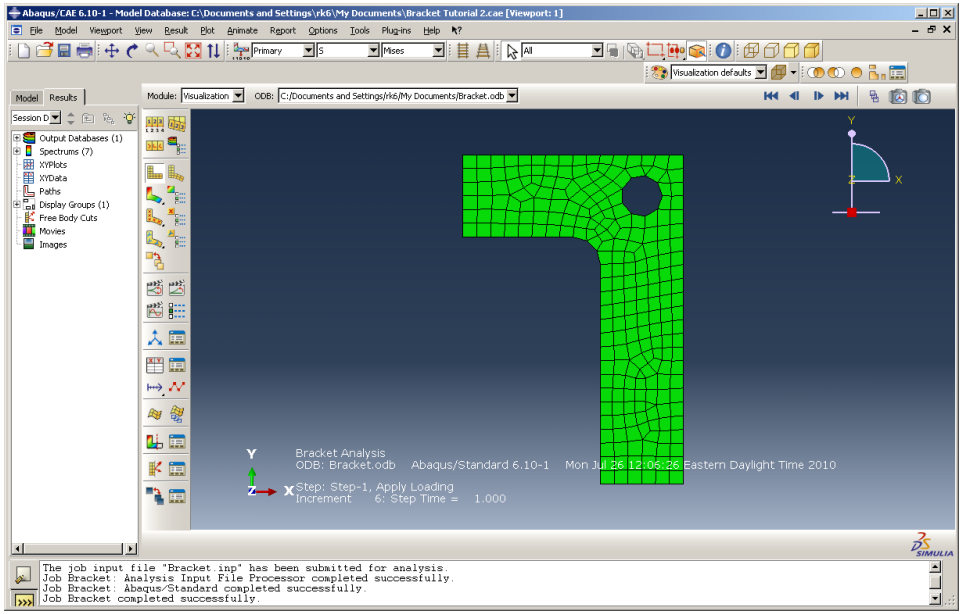
Submit Job



Job Manager

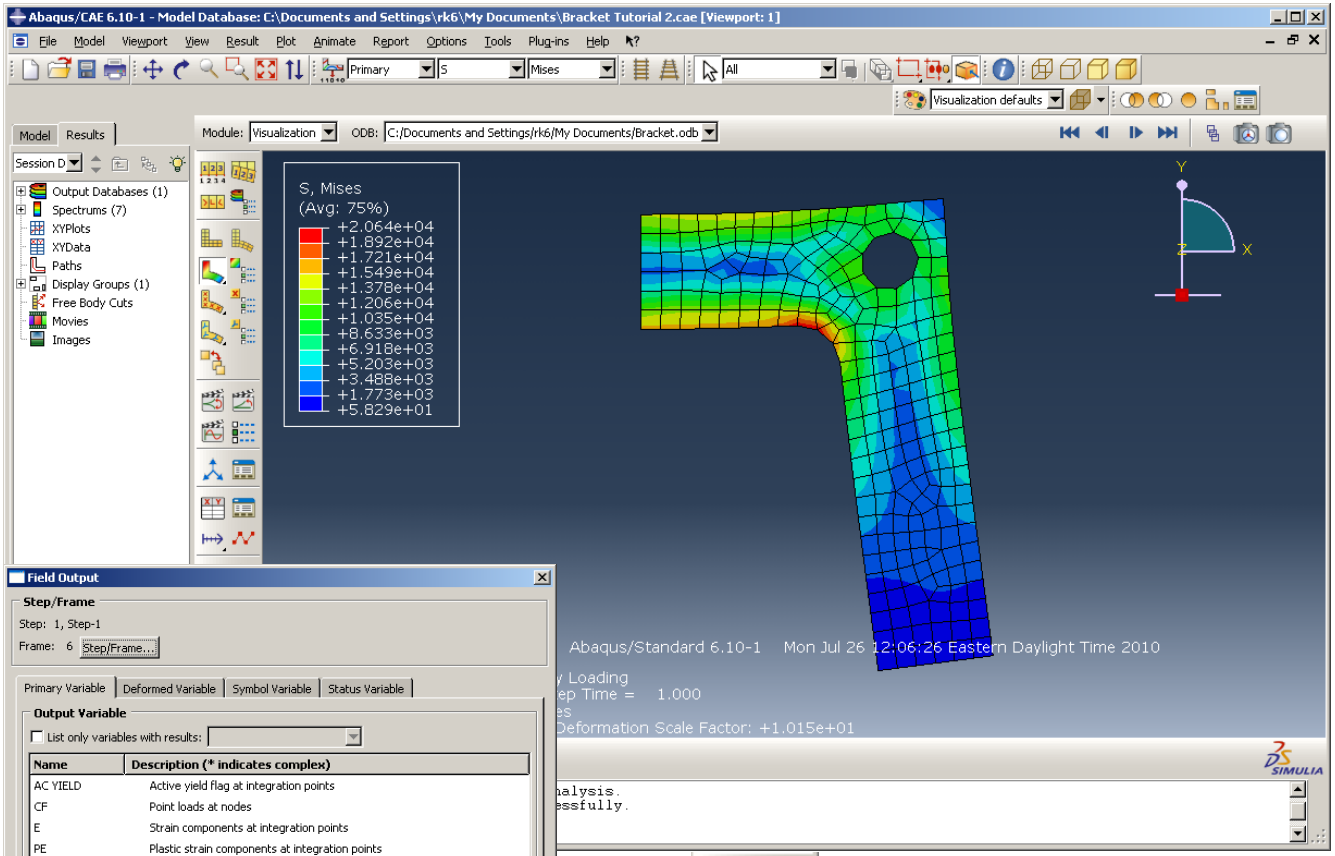
- Click **Job Manager button.**
- In *Job Manger* window pops up, check that your job is selected, then click **Submit.**
- To run your model in Unix Server, click **Write Input.** It takes few seconds to write "job name.inp".
- Then:
  1. Save you job
  2. Close ABAQUS/CAE
  3. Type *"abaqus job = job name"*
  4. Enter↵
  5. Then, go to slide 18-Result (b) for visualizing results
- Under *Status*, it will read*:*
  1. *Submitted*
  2. *Running*
  3. *Completed*
- Click **Results.**
- The Visualization module will run and the part will be shown. It should look like the image to the right.

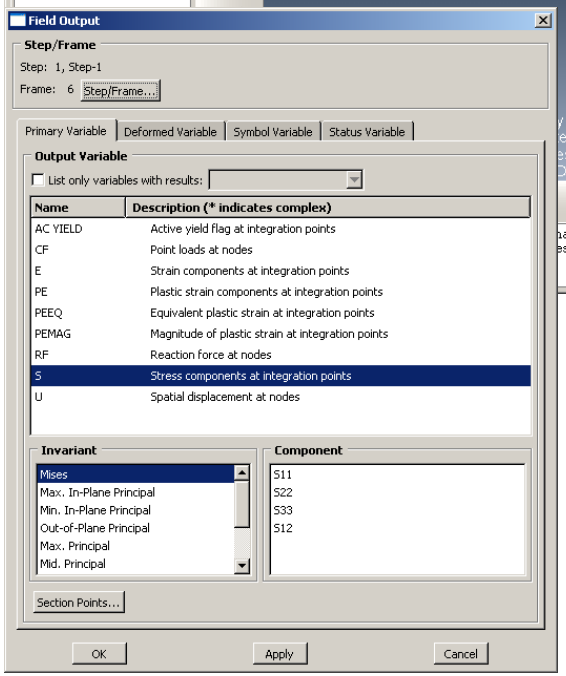


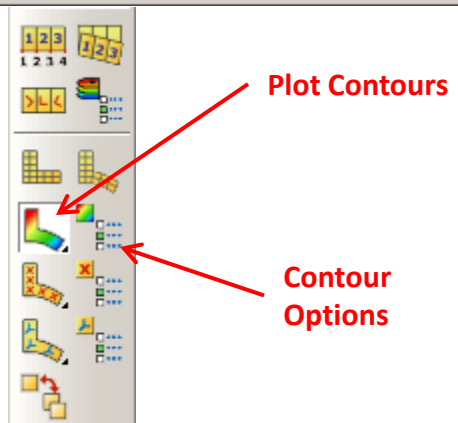

Bracket Analysis
ODB: Bracket.odb Abaqus/Standard 6.10-1
Step: Step-1, Apply Loading
The job input file "Bracket.inp" has been submitted for analysis.
Job Bracket: Analysis Input File Processor completed successfully.
Job Bracket: Abaqus/Standard completed successfully.
Job Bracket completed successfully.

Results (a)-Visualization

- Click the **Plot Contours** button.
- A colorful plot of Von Mises stresses appears.
- Color control can be adjusted by clicking the **Contour Options** button and adjusting parameters.
- To select the scalar field quantity plotted, in the *Menu Bar,* select **Result, Field Output,** then choose the stress component you wish to plot, e.g., S11, or U1.
- Click **OK.**
- Strains, Spatial Displacements, etc., can be selected through **Field Output.**

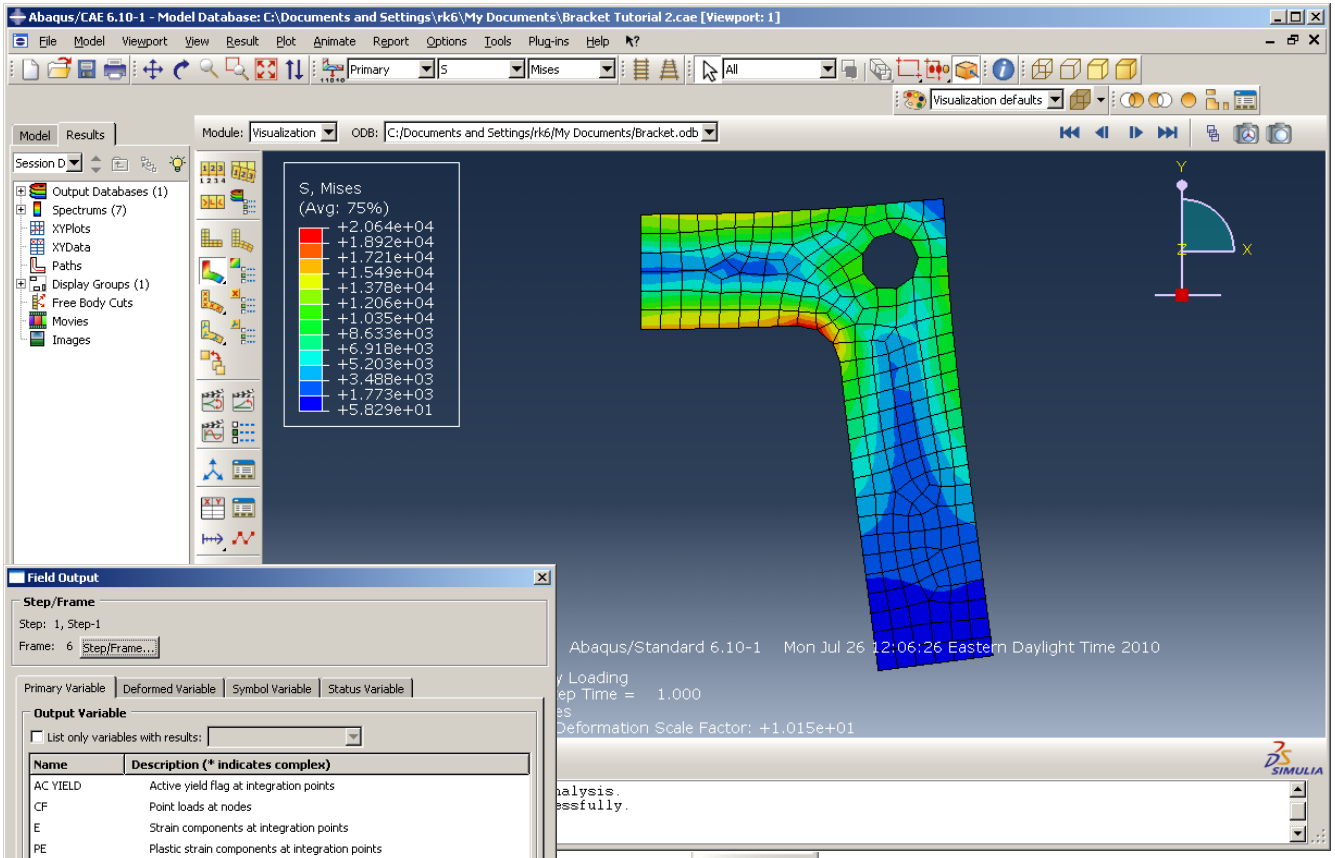


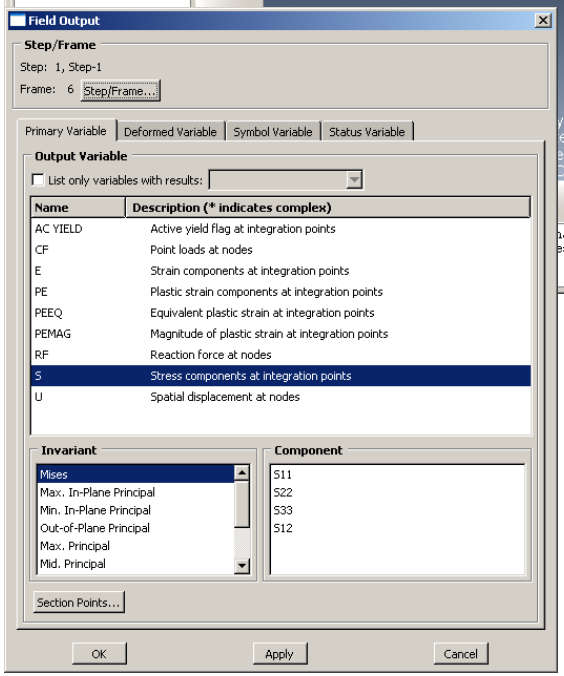




Results (b)-After Run Model on Unix Server

- Run ABAQUS/CAE or ABAQUS/VIEWER.
- Open “job name.odb”.
- Click the **Plot Contours** button.
- A colorful plot of Von Mises stresses appears.
- Color control can be adjusted by clicking the **Contour Options** button and adjusting parameters.
- To select the scalar field quantity plotted, in the *Menu Bar*, select **Result, Field Output**, then choose the stress component you wish to plot, e.g., S11, or U1.
- Click **OK.**
- Strains, Spatial Displacements, etc., can be selected through **Field Output.**





Plot Contours

Contour Options